

Soldermask Opening to Prevent Delamination

This application is a Divisional of U.S. Application No. 10/095,329, filed March 11, 2002, which is a Continuation of U.S. Application No. 09/417,491, filed October 13, 1999, now U.S. Patent No. 6,356,452, both of which are incorporated herewith.

Technical Field of the Invention

The present invention relates generally to printed circuit boards production and, in particular, to the processes utilized to create printed circuit boards.

10

Background of the Invention

A typical printed circuit board is a flat board that provides support and electrical interconnection between microchips and other electronic components. The base of the board can be made of reinforced fiberglass or plastic, and the electronic components are electrically interconnected by conductive pathways. There are several ways to make printed circuit boards. One method entails bonding a conductive foil, such as copper, over the base. A conductive pattern is then formed in the conductor. One method of patterning the metal layer uses a negative image of the desired circuit pattern and a photo resist layer. The photo resist is activated using the image such that selected areas of the photo resist can be removed. An etch process is then performed to remove the photo resist that was not activated and the underlying metal layer, leaving behind the conductive pathway pattern.

Today, most printed circuit boards are composed of several sheets or layers. A multi-layer printed circuit board may be fabricated from several composite sheets, each comprising a substrate of insulating material and a layer of metal, such as copper, attached to one surface of the substrate using a resin. A desired conductive pathway pattern is then provided in the metal layer, as explained, and multiple layers of insulating material and metal conductor layers can be fabricated. A soldermask layer can be provided over the top level of conductor to control areas exposed to a soldering

process. A finished printed circuit board can then put through an assembly process where it is populated with various electrical components.

Delamination, or separation, problems have been discovered in the printed circuit board assembly industry regarding processes similar to the one described above.

- 5 These problems are a result of the various environmental stresses inherent to assembly procedures. For example, delamination of the soldermask layer can occur when moisture retained by a board's constituent components are exposed to post-production assembly processes performed at high temperatures.

- For the reasons stated above, and for other reasons stated below which will
10 become apparent to those skilled in the art upon reading and understanding the present specification, there is a need for a multi-layer printed circuit board assembly that reduces the likelihood of delamination.

Summary of the Invention

- 15 The above mentioned problems with circuit board assemblies and other problems are addressed by the present invention and which will be understood by reading and studying the following specification.

- In one embodiment, a circuit board comprises a base layer, and a conductive layer located on a first side of the base layer. The conductive layer is patterned into
20 conductive traces. A soldermask layer is located on a surface of the conductive layer and a region of the first side of the base layer which is not covered by the conductive layer. The soldermask layer is provided with a first plurality of openings to expose the base layer to provide ventilation, and wherein the soldermask layer is further provided with a second plurality of openings to expose the conductive traces.

- 25 In another embodiment, a circuit board assembly comprises a base layer, and a conductive layer located on top of the base layer. The conductive layer is patterned to form conductive traces. A soldermask is attached over the conductor layer and the base layer. The soldermask layer includes a first plurality of openings to expose a top surface

of the base layer and a second plurality of openings to expose areas of the conductors. An integrated circuit is also attached to a first side of the circuit board.

Brief Description of the Drawings

5 Figure 1 is a top view of one embodiment of a simplified circuit board according to the present invention;

 Figure 2 is a cross section view of the circuit board shown in Figure 1;

 Figure 3 is a top view of an embodiment of a detailed circuit board according to the present invention; and

10 Figure 4 is a cross section view of the circuit board shown in Figure 3.

Detailed Description of the Invention

 In the following detailed description of the invention, reference is made to the accompanying drawings which form a part hereof, and in which is shown, by way of
15 illustration, specific embodiments in which the invention may be practiced. In the drawings, like numerals describe substantially similar components throughout the several views. These embodiments are described in sufficient detail to enable those skilled in the art to practice the invention. Other embodiments may be utilized and structural, logical, and electrical changes may be made without departing from the scope
20 of the present invention. The following detailed description is, therefore, not to be taken in a limiting sense, and the scope of the present invention is defined only by the appended claims, along with the full scope of equivalents to which such claims are entitled.

 Printed circuit boards, as described above, can be fabricated using multiple
25 layers of different materials. These materials can be insulators, conductors or other materials used to provide benefits during processing such as soldermasks. Because these materials differ in composition, they behave differently to environmental conditions such as temperature and humidity. Of particular interest to the present

invention is the retention of water, or other liquids, by insulating material used in fabrication of circuit boards.

Soldermasks, as known to those skilled in the art, are used to mask areas of a conductor to prevent solder from coating the conductor. That is, some areas of the
5 conductor may be required to be coated in solder, while other areas need to be protected from solder. By controlling the location of solder, conductive shorts can be prevented.

The soldermask layer can be screen printed over the conductor. The soldermask includes openings to expose selected areas of the conductor. It is desired that the soldermask remain adhered during processing. If the soldermask layer delaminates
10 from the conductor, conductive shorts or leakage paths can be formed.

The present invention provides a technique for reducing delamination of a soldermask layer of a circuit board which results from out-gassing of layers of the circuit board. Figure 1 is a top view of a simplified circuit board 100 according to the present invention. The circuit board includes a conductor trace 110 and a soldermask
15 used to shield the conductive layer from potential shorts by exposing only selected areas of the conductor via conductor openings 130. The soldermask also includes ventilation openings 140 to allow underlying layers of the circuit board to vent gases during processing without producing adverse side effects to the adhesion of the soldermask. The details of the vent openings, or holes, are described in greater detail below.

20 Figure 2 illustrates a cross-section view of the circuit board of Figure 1. The soldermask 120, or solder resist, allows access to the conductor 110 via conductor openings 130. Unlike prior soldermasks, the present invention provides an opening, vent holes 140 to expose a base layer 150. These openings allow for ventilation of the base layer. The base layer can be comprised of any electrically insulating material. In
25 one embodiment, the base material is Mitsubishi HCL832. The base layer can be fabricated using multiple layers, and is not limited to a single material. The soldermask can be either a sheet of insulating material which is adhered to the top of the base layer and conductor or a conformal coating. In one embodiment, the soldermask is a sheet of Taiyo AUS5 which is adhered to the board using a screen printing operation. Likewise,

the conductor traces can be any suitable conductive, or semi-conductive material such as copper.

As explained herein, the ventilation holes are provided in the soldermask layer to allow the base layer to out-gas. As such, the vent holes are located in a region of the circuit board which do not contain conductors. Because the soldermask in one embodiment is a sheet of material, the ventilation holes are patterned into a plurality of holes. That is, an array of holes is preferred to providing one large hole. The location and number of the ventilation holes will be dependant upon the design of the circuit board. In general, maximizing the cumulative surface area of ventilation holes in the locations not containing conductors is desired.

Figure 3 provides a top view of another embodiment of a circuit board 300 according to the present invention, and Figure 4 provides a cross-sectional view of the circuit board of Figure 3. The circuit board includes a conductive grid array 360 for providing an interconnect for circuit components, such as but not limited to resistors, capacitors, inductors, and integrated circuits. The soldermask layer 320 includes first openings to provide access to the grid array conductors, and second opening 340 to expose the base layer. These openings are generally provided along outside edges of the circuits board.

An integrated circuit die assembly is fabricated using the circuit board 300, as illustrated in Figure 4. The die assembly is comprised of the circuit board base layer 450 with a soldermask layer 320 concealing a conductive layer 410. A rectangular shaped opening, or aperture 310 is provided in the board. The opening passes through the soldermask layer, the conductive layer and the base layer. This opening provided access to an integrated circuit die, as explained below. A plurality of bond pads are located in regions 330 of the circuit board outside the aperture. A plurality of solder bump conductor openings 360 are located on the same side of the circuit board as the bond pads.

The die assembly comprises the integrated circuit die (IC) 460 which has its top side affixed to the bottom side of the circuit board. The IC can be affixed to the board

with an adhesive 470. The adhesive is applied to the IC's top side such bond connections provided on it's top side are exposed and fully accessible through aperture 310 when attached to the base layer. The IC can then be wire bonded through the opening to the bond pads provided in locations 330. The IC die and wire bonds are then
5 encapsulated with a non-conductive encapsulant 480 to provide an environmental seal.

Again, the soldermask layer included ventilation openings 340 to allow the underlying layers to expel gases without adversely affecting the adherence of the soldermask layer. This is of particular interest when moisture retained by a board's constituent components (e.g. a soldermask layer or a base layer) are exposed to high
10 temperatures delamination may occur in regions of the circuit board.

Conclusion

The invention allows gases trapped between layers of a multi-layered printed circuit board to escape, in an effort to prevent delamination. A circuit board has been
15 described which includes openings in a particular layer(s) of the multi-level printed circuit board. The present invention describes an improved multi-layer circuit board that is more resistant to delamination problems. An embodiment of a multi-layer circuit board according to the present invention comprises a base layer, a conductive layer and a soldermask. The soldermask layer has two primary sets of openings. One of these
20 sets are the vent openings, or vent holes, that expose the base layer to provide ventilation so that gases may escape. The circumference, location and depth of the holes are dependent upon the nature of the application. This invention allows a plurality of layers, such as the soldermask layer, to be used in printed circuit board applications, while preventing delamination throughout the board's surface. The reduction in
25 delamination offers the added benefit of allowing for improved handling of the circuit board, particularly during various processing stages.

Although specific embodiments have been illustrated and described herein, it will be appreciated by those of ordinary skill in the art that any arrangement which is calculated to achieve the same purpose may be substituted for the specific embodiment

shown. This application is intended to cover any adaptations or variations of the present invention. For example, by creating an avenue for gases to escape, the soldermask will remain attached at the board's perimeter, leaving a flat surface to clamp onto at the end of the assembly process. This makes further processing to the printed circuit board

5 possible.